



PROTEUS PCB DESIGN

Proteus PCB Design combines Schematic Capture and PCB Layout modules to provide a powerful, integrated and easy to use suite of tools for professional PCB design.

Productive PCB Design Tools

The Proteus PCB Design product range is a professional's choice for modern PCB Layout. With over 25 years of continuous development and innovation, our focus remains on adding functionality while maintaining a simple, clean user interface and tight integration with the schematic design.

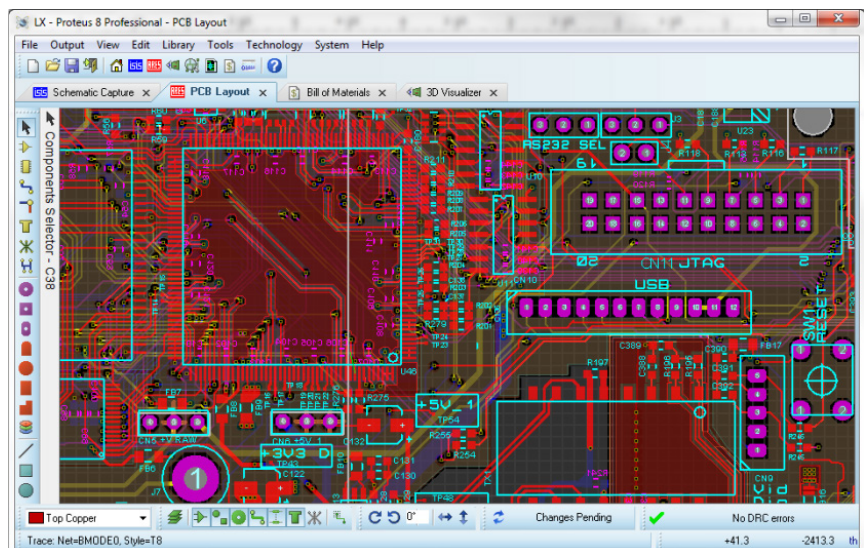
Proteus consists of a single application framework with several modules (Schematic, BOM, Layout, 3D Viewer, etc.). Each module opens in a tab within the application window but can then be dragged and dropped to create additional windows and enable side-by-side viewing. This allows you to not only work with the traditional schematic and layout windows but also to combine or split off additional modules according to the work you are doing at a particular time. For example, Schematic and BOM for report generation, Layout and 3D Viewer for verification and so on.

The main goal of the unified application framework is to improve and better support the user workflow. Changes made on the schematic during design will reflect visually in the layout and you can review and confirm pending changes or

go back and edit further on the schematic. Similarly, annotation changes made on the PCB can also feed back to the schematic. For example, the PCB can be automatically re-annotated after placement (e.g. top left to bottom right) and the schematic module will update accordingly. This puts control over the design process firmly in the hands of the engineer.

Labcenter's focus is on functionality and value for our customers. We are continually adding new features and tools to the Proteus

design Suite to ensure we have the features demanded by modern design techniques. To ensure our customers receive the best value, we equip all Proteus PCB products with a rich set of core functionality including a world class shape based autorouter as standard. More advanced features such as high speed length matching can then be added in the higher product levels as required. Upgrading your Proteus design Suite is just a matter of paying the difference as we provide a 100% credit for your previous purchase.



"PCB design is our business. We review PCB layout software on an ongoing basis and Labcenter has topped the list for the last 10 years. Certainly the most productive and very, very affordable. We have licences for other very expensive products but they don't get much use."

Donald Kay - Don Alan Pty



PCB LAYOUT

3D Visualization with import and export of STEP/IGES models.



Automatically tune route lengths via a simple select and match user interface.



BOARD CONSTRAINTS

Design rules in Proteus can be as simple or as detailed as you need. You start with a board global rule which defines a base set of clearances. You can then add additional rules for specific layers or for specific sets of signals which will override the global clearances where appropriate. Finally, you can draw out a design rule room on the PCB and set specific clearances in that area. This is particularly useful for things like BGA escape areas or fine pitch SMD fanout regions. Once your board constraints are set up the DRC system will notify you of any violations via a simple click to zoom list viewer, making it easy to find and fix problems during board layout.



IMPORT PARTS

Creating library parts for schematic and layout can be tedious and error prone and diverts the engineer from the task of actually making the product. Fortunately, there are now a variety of third party websites with catalogues of literally millions of library parts available for import. We've integrated these web results directly into the library pick form where a double click on a result will seamlessly download and import the part. There is also a manual import part dialogue designed to work with multiple tools such as Ultra-Librarian, SnapEDA and PCB Library Expert. Both schematic symbols and PCB footprints are imported at the same time along with the pin mapping between them and often the accompanying STEP file for 3D Visualization.



ROUTE PLACEMENT

During track placement the route will follow the mouse wherever possible and will intelligently move around obstacles while obeying the design rules for the project. Curved tracks can be laid down simply by pressing the CTRL key during route placement. Teardrops can be added globally to track/pad connections via a simple dialogue form. High speed signals can be length matched either to each other or to a specified distance. This includes the ability have track segments in multiple matchgroups which is needed for routing topologies like the DDR3 fly-by configuration. Finally, a shape based autorouter is included as standard to quickly achieve routing completion on non-critical nets.



BOARD SETUP

The layer stackup manager lets you specify the number of cores in the board, the thickness of each layer and the order in which they are assembled to make the final PCB. Proteus can then work out the legal, manufacturable set of default drill passes for the layout. You can add or remove drill passes over various layer ranges if you need to. When you are then routing a track and you place a via, the software can automatically choose the smallest via range from source layer to destination layer. It can also use the via depth in length matching calculations for high speed signals. Both the layer stackup information and the drilling table are then exported as part of the the Gerber X2 fileset for manufacture.

Re-use previously designed schematic & layout circuitry with Project Clips.



Use Assembly Variants to manage different product configurations.

